FACULTY OF MECHANICAL ENGINEERING TUL



# Introduction to Computational Fluid Dynamics (CFD)

## Lectures 1 and 2 (Part 2)

jan.kracik@tul.cz

# Agenda – Lecture 1

- Approaches to solving engineering problems
- CFD (What? Why? How?)
- Fundamental equations

# Investigation approaches

**Experimental investigation**

- Provides the most reliable information
- Can use both full-scale and small-scale tests
- Not free from errors

**Theoretical calculation**

- Usually a set of differential equations
- Solutions exist only for a narrow range of practical problems

**Numerical calculation (CFD)**

- Finite number of domain elements (discretization)
- A set of algebraic equations
- Solutions exist for almost every practical application

# Strengths and weaknesses of the theoretical approach

**Advantages (Strengths)**

- Speed
- Low cost
- Information completeness
- Ability to simulate both ideal and real conditions

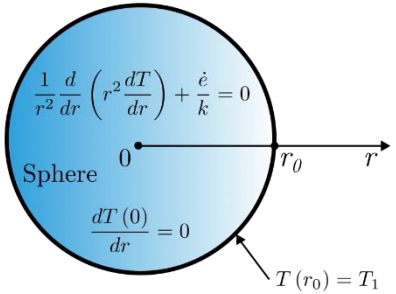
**Disadvantages (Weaknesses)**

- Mathematical model
- Problem complexity

**Remember: Experiment leads, computation follows.**

# Why use numerical methods?

**Problem: Heat conduction in a sphere**

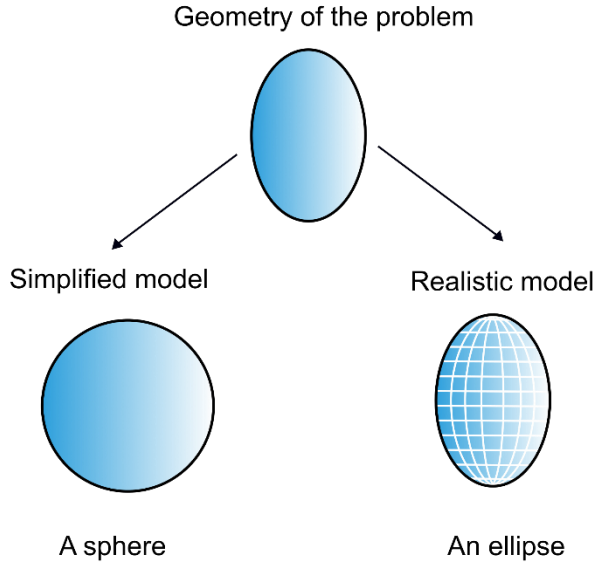



**Governing differential equation:**

$$\frac{1}{r^2}\frac{d}{dr}\left(r^2\frac{dT}{dr}\right)+\frac{\dot{e}}{k}=0$$

**Solution:**

$$T(r)=T_1+\frac{\dot{e}}{6k}\left(r_0^2 - r^2\right)$$



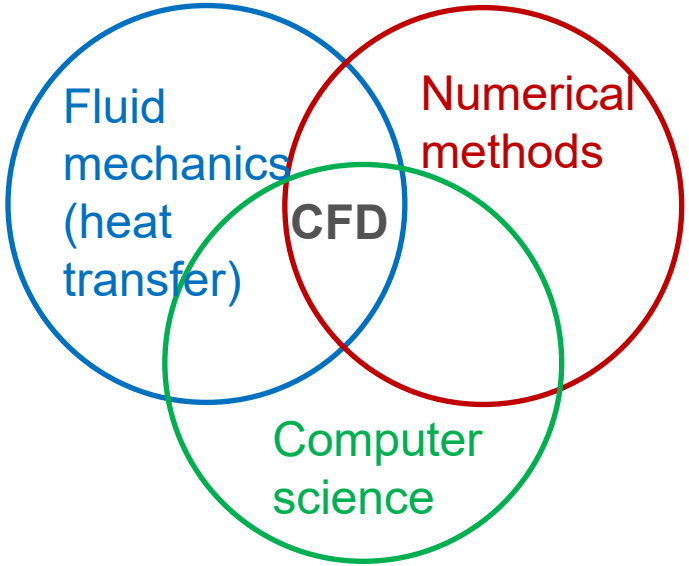

Exact (analytical) solution of model, but crude solution of actual problem

Approximate (numerical) solution of model, but accurate solution of actual problem

# Why prefer numerical methods over analytical ones?

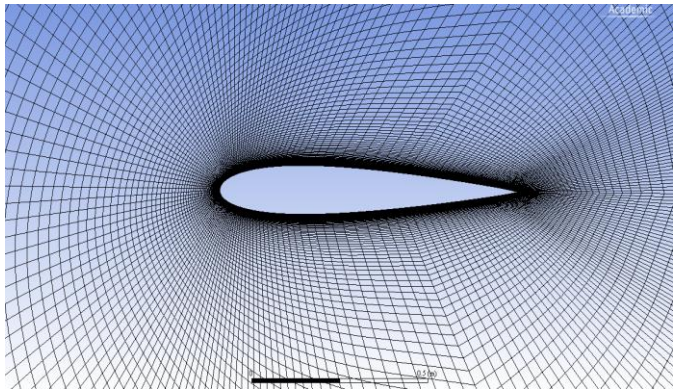
- **Limitations** (geometry, variable HTC, temperature-dependent k, …)
- **Better modeling** (“approximate” solution of a realistic model is usually more accurate than the “exact” solution of a crude mathematical model)
- **Flexibility** (parametric studies to answer “what-if” questions)
- **Complications** (even when analytical solutions are available, they can be intimidating)
- **Human nature** (ready availability of high-powered computers with sophisticated software packages)

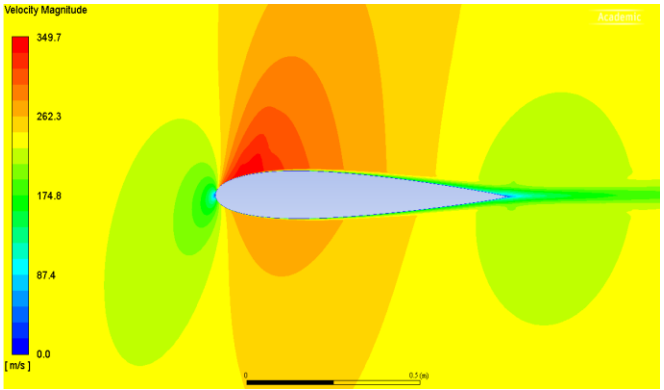
# What is CFD?


Fluid mechanics (heat transfer)
Numerical methods
CFD
Computer science

# What is CFD?

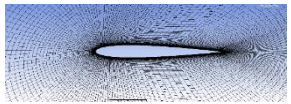
- Numerical analysis and computers come into play.
- Differential equations → Algebraic equations → solution
- The application of CFD to practical problems is often limited by the computational power available.



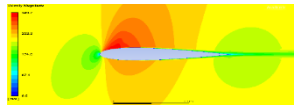



# The black box idea

- This applies especially to commercial packages (Ansys Fluent, CFX, STAR-CCM+).
- More sophisticated codes are also available (OpenFOAM).
- The user provides the inputs (geometry, mesh, boundary conditions, material properties, solver settings).
- Turn the crank and get the results (colour pictures).



User inputs → **Black box** → Color pictures and other results



# Set of fundamental equations

1. **Conservation of mass**

$$\frac{\partial \rho}{\partial t} + \nabla \cdot [\rho \mathbf{U}] = 0 \qquad (1)$$

2. **Conservation of momentum**

$$\frac{\partial [\rho \mathbf{U}]}{\partial t} + \nabla \cdot \{\rho \mathbf{U}\mathbf{U}\} = [\mathbf{f_s}] + [\mathbf{f_V}] \qquad (2)$$

3. **Conservation of energy**

$$\frac{\partial (\rho e_{total})}{\partial t} + \nabla \cdot [\rho e_{total} \mathbf{U}] = \mathbf{f_s} \cdot \mathbf{U} + \mathbf{f_V} \cdot \mathbf{U} - \nabla \cdot [\mathbf{q}] \qquad (3)$$

Note:

$$e_{total} = e_{internal} + \frac{1}{2}\mathbf{U} \cdot \mathbf{U} \qquad (4)$$

$$\mathbf{q} = -k \nabla T \qquad (5)$$

# Set of fundamental equations

1. **Conservation of mass**

$$\frac{\partial \rho}{\partial t} + \nabla \cdot [\rho \mathbf{U}] = 0 \tag{6}$$

2. **Conservation of momentum**

$$\frac{\partial [\rho \mathbf{U}]}{\partial t} + \nabla \cdot \{\rho \mathbf{U}\mathbf{U}\} = [\mathbf{f_s}] + [\mathbf{f_V}] = \nabla \cdot \boldsymbol{\sigma} + [\rho \mathbf{g}] = -\nabla p + \nabla \cdot \boldsymbol{\tau} + [\rho \mathbf{g}] \tag{7}$$

3. **Conservation of energy**

$$\frac{\partial (\rho e_{total})}{\partial t} + \nabla \cdot [\rho e_{total} \mathbf{U}] = \mathbf{f_s} \cdot \mathbf{U} + \mathbf{f_V} \cdot \mathbf{U} - \nabla \cdot [\mathbf{q}] = -\nabla \cdot [p\mathbf{U}] + \nabla \cdot [\boldsymbol{\tau} \cdot \mathbf{U}] + [\rho \mathbf{g}] \cdot \mathbf{U} + \nabla \cdot [k \nabla T] \tag{8}$$

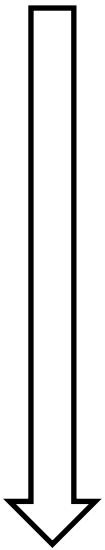
# Summary – Lecture 1

- CFD can be a handy tool (What? Why? How?)
- Governing equations: **mass**, **momentum** (Newton's 2nd law), and **energy** (1st law of thermodynamics)

# Agenda – Lecture 2

- Numerical procedure
- Some CFD applications
- Turbulence in CFD

# Numerical simulation procedure



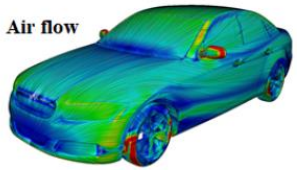

1. Mathematical model
2. Physical model (steady, incompressible, etc.)
3. Domain discretization → Numerical mesh
4. Equation discretization → A set of algebraic eqs.
5. Solution method and boundary condition settings
6. Convergence criteria
7. Solution calculation
8. Results interpretation and presentation

Pre-processing

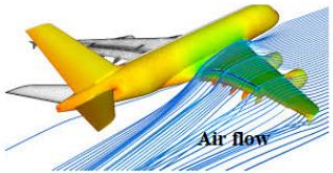
Processing or solving

Post-processing

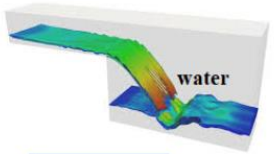
# Applications of CFD



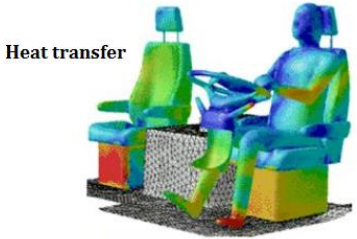

Automobile



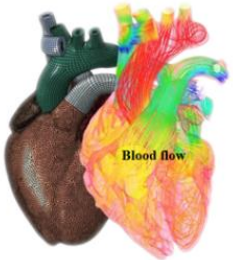

Aerospace




Water fall

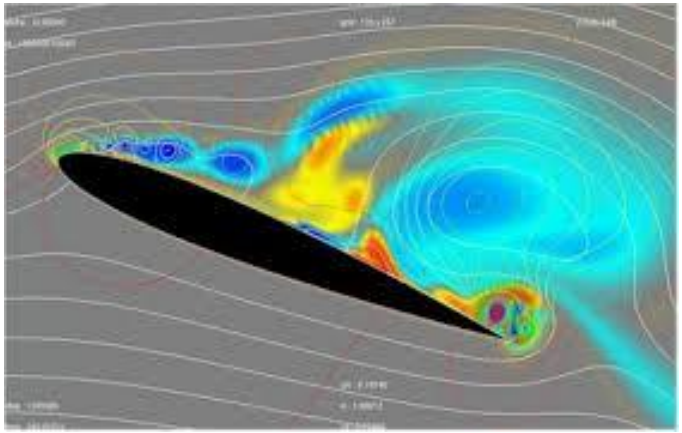



Ventillation



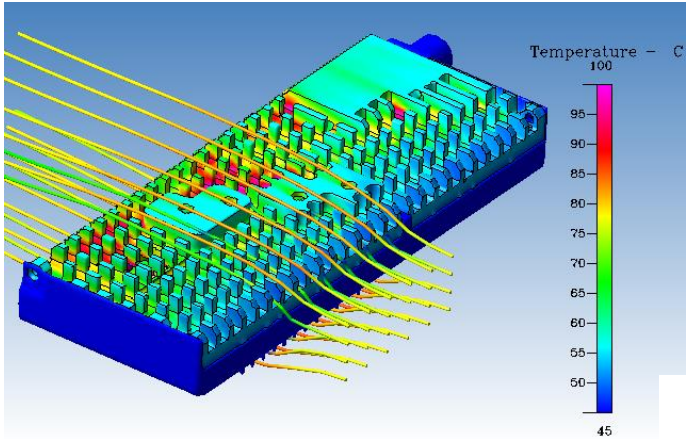

Biomedical

Aerospace/Energy

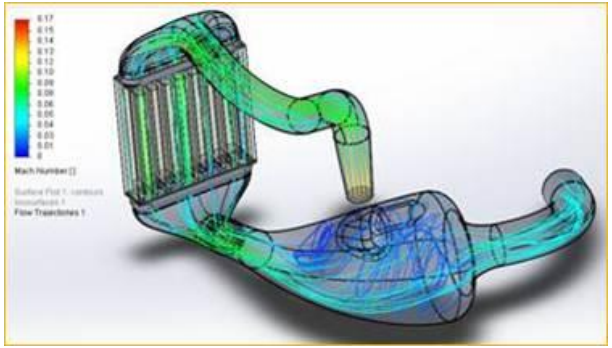


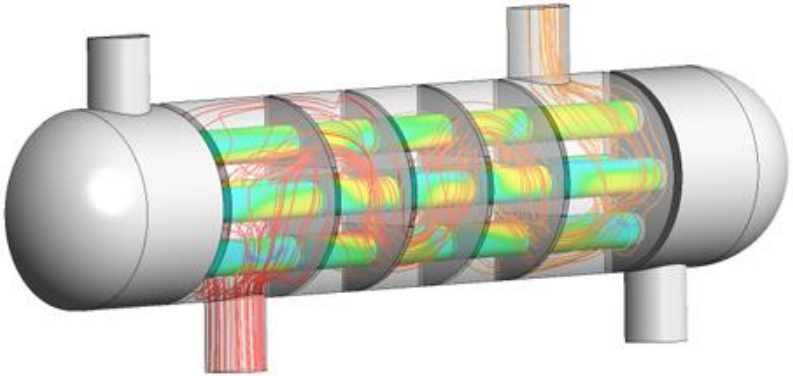
Source: https://cfdflowengineering.com/scope-of-cfd-modeling-career-and-job-opportunities/

# Applications of CFD (2)




**Heat transfer**

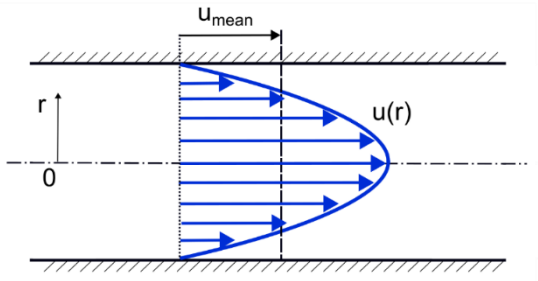




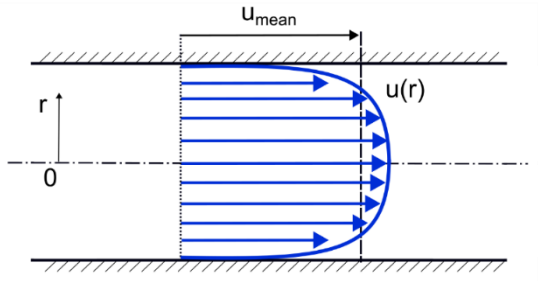
# Treatment of turbulence in CFD

- Flow can be laminar, turbulent (more common), or transitional (difficult to solve).
- Most flows in practice are turbulent.
- Laminar solutions are almost exact (limited only by mesh resolution and BCs).
- Fully resolving complex turbulent flows is challenging and not yet feasible.

**Laminar profile**



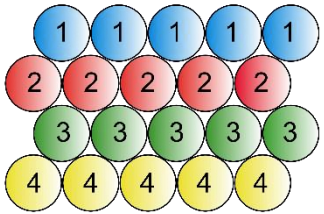

**Turbulent profile**



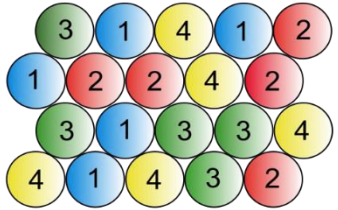
# Treatment of turbulence in CFD (2)

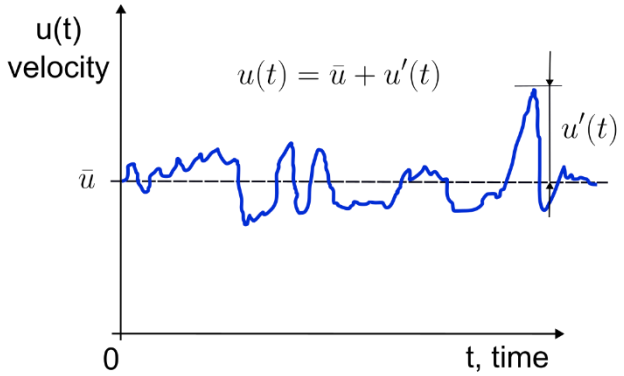
- Direct Numerical Simulation (DNS) is not practical for engineering problems.
- It would require a very fine mesh to capture all turbulent motions.
- Therefore, we must rely on experiments and empirical correlations.

Before turbulence




After turbulence



# Treatment of turbulence in CFD (3)

- Turbulent flows are characterized by rapid, random fluctuations of swirling regions (eddies).
- We need to capture these turbulent structures somehow.
- One option is to draw inspiration from molecular motion.
- In laminar flows, shear stress is caused by molecular viscosity.
- In turbulent flows, this shear stress is still present, and additional stresses arise from the turbulent fluctuations.

$$\boldsymbol{\tau}_{\text{total}} = \boldsymbol{\tau}_{\text{laminar}} + \boldsymbol{\tau}_{\text{turbulent}} \quad (9)$$

# Treatment of turbulence in CFD (4)

- **The laminar component** of the total shear stress can be expressed as:

$$\boldsymbol{\tau}_{\text{laminar}} = -\mu \frac{\partial \bar{U}}{\partial r} = -\mu \frac{\partial \bar{U}}{\partial y} \tag{10}$$

- In an analogous manner, we can express **the turbulent component**:

$$\boldsymbol{\tau}_{\text{turbulent}} = -\mu_t \frac{\partial \bar{U}}{\partial r} = -\mu_t \frac{\partial \bar{U}}{\partial y} \tag{11}$$

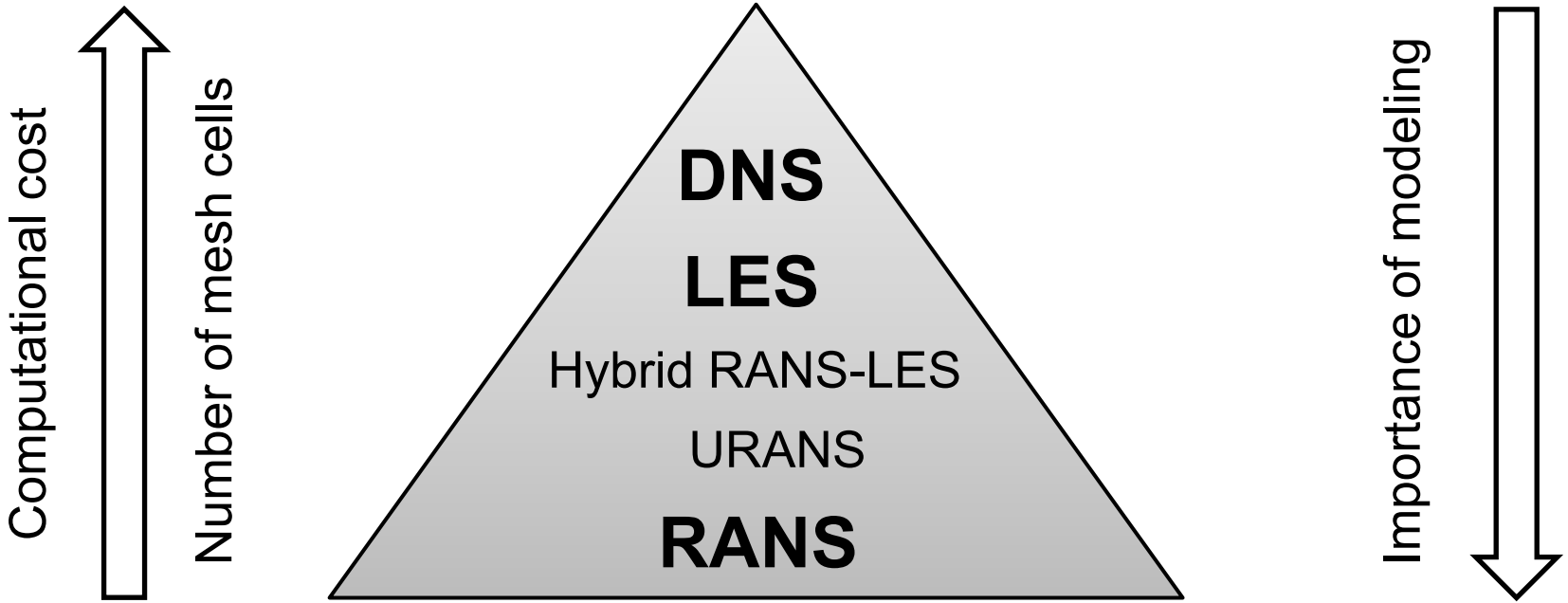
- The problem is that we do **not know $\mu_t$**: unlike μ, it is not a material constant but a property of the turbulent flow.

- Note that the turbulent shear stress is often also expressed as: $\boldsymbol{\tau}_{\text{turbulent}} = -\rho\overline{u'v'}$ (12)

# Treatment of turbulence in CFD (5)

- Most simulations require a turbulence model (allowing a coarser mesh).
- No universal model exists for all turbulent flows.
- Turbulence models aim to represent the effect of turbulence through additional terms or equations.
- Models try to capture the mixing and diffusion caused by turbulent eddies.
- CFD results are only as good as the turbulence model used.

# Turbulence models in CFD



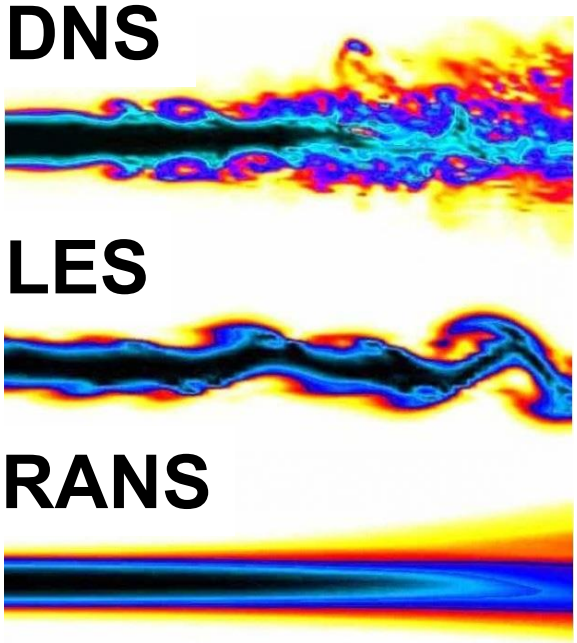

DNS - Direct Numerical Simulation
LES - Large Eddy Simulation

RANS - Reynolds-Averaged Navier-Stokes
URANS - Unsteady (transient) RANS

# Turbulence models in CFD (2)

Computational cost

Number of mesh cells




Importance of modeling

DNS - Direct Numerical Simulation
LES - Large Eddy Simulation

RANS - Reynolds-Averaged Navier-Stokes
URANS - Unsteady (transient) RANS

# Summary – Lecture 2

- Computational process step by step (from pre- to post-processing)
- Some applications
- Treatment of turbulence

# Thank you.